P517296.PDF [Page: 3 of 50]

Copy No. 12

DEFENCE R&D DÉFENSE

Development of a Model Generator and Graphical User Interface for the VVES Programs

Polak, M.F., Jiang, L., MacAdam, T.A., Chernuka, M.W.

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No. W7707-008286/001/HAL

Defence R&D Canada

Contractor Report
DREA CR 2001-143
January 2002

National Defence Défense nationale

Canada

P517296.PDF [Page: 4 of 50]

104-06258/

Copy No: _____

Development of a Model Generator and Graphical User Interface for the VVES Programs

Polak, M.F., Jiang, L., MacAdam, T.A., Chernuka, M.W.
Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No. W7707-008286/001/HAL

Defence Research Establishment Atlantic

Contractor Report
DREA CR 2001-143
January 2002

7296.PDF [Page: 5 of 50]

Authors

M.F. Polak L. Jiang T.A. MacAdam M.W. Chernuka

Contract Scientific Authority

D.C. Stredulinsky J.P Szabo

Approved by

J.L. Kennedy

Head, Warship Signatures & Safety

Approved for release by

Kirk Foster

Chair/DRP

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

© Her Majesty the Queen as represented by the Minister of National Defence, 2002

© Sa majesté la reine, représentée par le ministre de la Défense nationale, 2002

P517296.PDF [Page: 6 of 50]

Abstract

The VVES (Vibration of Visco-elastic and Elastic Structures) suite of programs is a group of Matlab codes for analysis and design of engine isolation systems composed of engine, raft and ground which are connected by visco-elastic isolators. Among these programs, VVESANALYSIS allows the dynamic characteristics of isolators to be described by frequency-dependent complex material properties. The VVESDYNSTIFF and VVES4POLE require input of the directly measured dynamic stiffnesses and the 4 pole parameter matrices, respectively.

In order to simplify the construction of input files and provide an alternate method to view the VVES analysis results, a model generator and graphical user interface GUI for the VVES programs has been recently developed called VIMGEN (Vibration Isolation Model GENerator). This model generator and GUI were developed for the Windows environment using the DREA Hierarchical Object Oriented Developer's toolkit (HOOD). Details of this development are documented in this report. The reader of this report is assumed to have a reasonable familiarity with the VVES programs and the HOOD toolkit's core components.

Résumé

La famille de logiciels VVES (*Vibration of Visco-elastic and Elastic Structures* : vibrations des structures visco-élastiques et élastiques) regroupe des programmes MATLAB conçus pour analyser et concevoir des systèmes d'atténuation des vibrations des ensembles moteur-radier-coque solidarisés par des isolateurs visco-élastiques. Un de ces programmes, VVESANALYSIS, utilise les propriétés complexes des matériaux, lesquelles varient avec la fréquence, pour prédire les caractéristiques dynamiques des isolateurs. L'exécution de VVESDYNSTIFF requiert l'entrée des rigidités dynamiques mesurées directement et celle de VVES4POLE, des matrices quadripolaires de paramètres.

Afin de faciliter la création des fichiers d'entrée et fournir une autre méthode de visualisation des résultats de l'analyse réalisée par VVES, nous avons récemment élaboré, pour cette famille de programmes, le logiciel VIMGEN qui englobe un générateur de modèles et une interface graphique. (VIMGEN : *Vibration Isolation Model GENerator* ou générateur de modèles d'atténuation des vibrations). Nous avons élaboré ce générateur et cette interface graphique, pour l'environnement Windows, à l'aide des outils HOOD (*Hierarchical Object Oriented Developer's toolkit* ou trousse de producteur de logiciels hiérarchiques orientés objet) du CRDA. Le présent guide contient des renseignements sur cette réalisation. Le lecteur devra connaître raisonnablement bien les programmes VVES et les éléments centraux de la trousse logicielle HOOD.

7296 PDF [Page: 7 of 50]

This page intentionally left blank.

P517296.PDF [Page: 8 of 50]

Executive summary

Background:

Marine diesel engines are supported by mounts that are designed to provide both structural rigidity and vibration isolation. Structural rigidity is required in order to maintain alignment of connecting shafts and piping; and vibration isolation is required in order to minimize the vibrations generated by the engine from being transmitted to the rest of the vessel and beyond. The vibration isolation materials currently in service on Canadian vessels, natural rubber and chloroprene rubber, experience large changes in their mechanical properties when exposed to lubricating oil or fuel. Replacing these materials with alternative elastomers could prevent alignment problems due to mount degradation, as well as reduce maintenance costs by increasing the length of time between mount changes. Replacement mounts must have a higher resistance to hydrocarbons and they must provide optimal vibration isolation. The optimal design of such mounts from a vibration viewpoint is complicated by the complex constitutive characteristics of the elastomers used in the construction of the mounts, the static compression of the mounts, and also by the complex nature of the three-dimensional vibration motion of the engine. DREA has developed sophisticated software for modeling engine vibration isolation systems. This code, called VVES (Vibration of Visco-elastic and Elastic Structures), provides a powerful tool for characterizing the dynamic properties of vibration isolation systems. It was not convenient to use VVES for practical engineering problems due to the large number of input files it requires.

Principal Results:

The principal result arising from this work is the development of a computer program called VIMGEN (Vibration Isolation Model GENerator), which is a GUI based model generator and control program for running the VVES software and visualization of results of analysis of engine/raft/foundation systems. This report documents the development of the VIMGEN software.

Significance of Results:

A sophisticated computational tool has been developed that will be of value in the optimal design of new isolators for marine diesel engines. The GUI based tools developed in this work will make model generation and running of the VVES code much easier and faster, especially for the novice or infrequent user, and should reduce errors in model generation.

Polak, M.F., Jiang, L., MacAdam, T.A., Chernuka, M.W. 2002. Development of a Model Generator and Graphical User Interface for the VVES Programs. DREA CR 2001-143. Defence Research Establishment Atlantic.

17296.PDF [Page: 9 of 50]

Sommaire

Contexte:

Les diesels marins reposent sur des supports dont la fonction est double : assurer la rigidité structurale et amortir les vibrations. La rigidité structurale permet de maintenir l'alignement des arbres et des conduites, tandis que l'amortissement des vibrations permet d'isoler le bâtiment et l'environnement des trépidations du moteur. Or, les propriétés mécaniques des matériaux antivibratiles couramment utilisés dans les navires canadiens — le caoutchouc naturel et le néoprène — sont fortement altérées par le contact avec l'huile lubrifiante ou le carburant. Substituer ces matériaux par d'autres élastomères pourrait prévenir les désalignements causés par la dégradation des supports et, en allongeant leur vie utile, réduire les coûts d'entretien. Les blocs de remplacement devront mieux résister aux hydrocarbures et assurer l'amortissement optimal des vibrations. Du point de vue de l'atténuation des vibrations, la conception optimale de ces supports est compliquée par les caractéristiques constitutives complexes des élastomères utilisés dans leur fabrication, la compression statique des supports et la nature complexe des mouvements vibratoires du moteur dans les trois dimensions. Le CRDA a produit un logiciel perfectionné, VVES (*Vibration of Visco-elastic and Elastic Structures* : vibrations des structures viscoélastiques et élastiques), pour modéliser les systèmes antivibrations des moteurs. Ce logiciel est un outil puissant de caractérisation des propriétés dynamiques des systèmes antivibrations. Il n'était pas pratique de l'utiliser pour résoudre des problèmes concrets d'ingénierie à cause du nombre élevé de fichiers d'entrée qu'il exigeait.

Résultats principaux

Le résultat principal de ce travail est la création de VIMGEN (*Vibration Isolation Model GENerator* : générateur de modèles d'atténuation des vibrations), un logiciel qui combine un générateur de modèles à interface graphique ainsi qu'un programme de commande permettant d'exécuter le logiciel VVES, et la visualisation des résultats de l'analyse de l'ensemble moteur-radier-coque. Le présent document donne des information sur le logiciel VIMGEN.

Importance des résultats:

Nous avons élaboré un outil informatique perfectionné qui sera utile pour la conception optimale de nouveaux supports antivibrations des diesels marins. Ces outils qui exploitent une interface graphique élaborée au cours de ce travail faciliteront et accéléreront l'exécution du logiciel VVES, en particulier pour les utilisateurs débutants ou occasionnels, et devraient réduire le nombre d'erreurs dans la production de modèles.

Polak, M.F. et Jiang, L. 2002. *VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual.* DREA CR 2001-142. Centre de recherches pour la Défense Atlantique.

TABLE OF CONTENTS

TABLE OF CONTENTS ..... **V**

1. BACKGROUND **1.1**

1.1 INTRODUCTION 1.1

1.2 SYSTEM ARCHITECTURE 1.2

2. IMPLEMENTATION **2.4**

2.1 ATTACHMENT POINTS 2.4

2.1.1 Properties *2.5*

2.1.2 Serialization *2.6*

2.2 ENGINE 2.6

2.2.1 Properties *2.7*

2.2.2 Serialization *2.9*

2.3 RIGID RAFT 2.9

2.3.1 Properties *2.10*

2.3.2 Serialization *2.11*

2.4 BEAM RAFT 2.11

2.4.1 Properties *2.12*

2.4.2 Serialization *2.13*

2.5 ISOLATORS 2.15

2.5.1 Properties *2.15*

2.5.2 Serialization *2.18*

2.6 ISOLATOR LAYERS 2.19

2.7 FREQUENCY-DEPENDENT VISCO-ELASTIC MATERIAL PROPERTIES 2.19

2.7.1 Experimentally Measured Properties *2.19*

2.7.2 Havrilak-Negami Material Model *2.20*

2.7.3 Rational Polynomial Material Model *2.20*

2.7.4 Direct Complex Dynamic Stiffness *2.21*

2.7.5 4-Pole Parameter Matrices *2.22*

2.8 LUMPED MASS 2.22

517296.PDF [Page: 11 of 50]

2.8.1 Properties 2.22
2.8.2 Serialization 2.24
2.9 CONFIGURATION INFORMATION (VVESINFO NODE) 2.25
2.9.1 Properties 2.25
2.9.2 Serialization 2.25
2.10 VVES INPUT FILE CREATION 2.27
2.11 MODEL WIZARD 2.27
2.12 MATLAB SERVER 2.27
3. POST PROCESSING 3.29
3.1 CHARTING 3.29
3.2 ANIMATION 3.29
4. TESTING 4.30
5. REFERENCES 5.31
APPENDIX A: CALCULATION OF ISOLATOR ORIENTATION PLANE FROM...A-1
APPENDIX B: CALCULATION OF SCALING FACTOR OF YOUNG'S MODULUS FOR TAKING INTO ACCOUNT THE PRE-STRESS EFFEC.....B-1

P517296.PDF [Page: 12 of 50]

1. BACKGROUND

1.1 Introduction

In order to provide both structural rigidity and vibration isolation, marine diesel engines are supported by mounts which are normally made of visco-elastic materials, such as natural rubber and chloroprene rubber. Structural rigidity is required to maintain alignment of connecting shafts and piping, whereas vibration isolation is required to minimize transmission of vibrations generated by the engine. The analysis and optimum design of these engine mounts are often complicated due to the complex constitutive characteristics of the visco-elastic materials and the complex nature of the three-dimensional motion of the engine.

In order to provide a numerical tool for modeling the performance of vibration isolation mounts, a group of Matlab computer codes, named VVES (Vibration of Visco-elastic and Elastic Structures), have been developed at the University of British Columbia under the sponsorship of DREA [1]. The general system configuration analyzed by these programs is a two-stage isolation system. In this system, the engine is connected to a foundation raft through elastomeric isolators and the foundation raft itself is isolated from the hull of the ship by a second set of isolators, where the ship hull is considered rigid in the present version of the VVES programs. These VVES programs can be used to calculate various dynamic characteristics of the engine/raft/mounts systems, such as natural frequencies and free vibration mode shapes, frequency response functions, force and displacement transmissibility functions and quasi-static deformations due to gravity load.

Four independent Matlab programs are included in the VVES program suite, allowing for different ways of defining frequency-dependent properties of the engine mounts. The VVESANALYSIS program accepts frequency-dependent complex constitutive properties of the visco-elastic materials in terms of the shear and bulk moduli, and computes the condensed stiffness and mass matrices of the isolators by performing linear static finite element analyses. The VVESDYNSTFF and VVES4POLE programs perform dynamic analyses using either the dynamic stiffness components or the 4 pole parameters directly measured from experiments. The fourth program, VVESACTIVE, is similar to VVESANALYSIS, but is equipped with an active control algorithm

P517296.PDF [Page: 13 of 50]

for reducing the force transmissibility of the system. The first three programs are to be considered in the present contract.

Although the VVES programs provide a powerful tool for characterizing dynamic properties of vibration isolation systems, their use in practical engineering analyses is not convenient due to the large number of input data files they require. The main purpose of the present work is to develop an application to assist users in generating engine isolation system models for analysis by the VVES programs, seamlessly spawning these programs on the generated models, and providing a means to visualize the results of the analyses.

1.2 System Architecture

The Hierarchical Object Oriented Developer's Toolkit (HOOD-TK) [2] was chosen as the framework in which to implement the VVES model generator application. The HOOD toolkit is an ongoing project on behalf of DREA to create a standard system for developing unified, graphical user interface (GUI)-based visualization and simulation applications involving graphics, animations, and event-driven user interaction. The HOOD toolkit is an application framework based on a domain engineering approach to software development, with its main focus currently on modeling marine vessel architecture. HOOD benefited this project mainly in two ways: first, it provided a hierarchical, object oriented data format in which to model the isolation system structural components, and second, provided a framework upon which a custom application could be built to interact with the isolation system model.

In order to properly model the engine isolation system in HOOD, an object-oriented analysis of the system was undertaken to determine the main structural entities, their associated attributes and their interaction dependencies. From this analysis various HOOD object classes were developed to model the engine system (these objects are discussed in detail in Section 2). The encapsulated HOOD classes were arranged in a top-down hierarchical manner to form the engine system model. Wherever possible, such as in the case of isotropic material data and beam cross sectional data, existing HOOD classes were used instead of defining new classes.

The HOOD application framework expedited the creation of the basic MFC (Microsoft Foundation

Classes)-based multiple document interface (MDI) modeling application. The framework provided a great deal of existing functionality, such as the OpenGL model plotting and interaction capabilities as well as the model object message passing and serialization mechanisms.

Customization of the basic application included the addition of new HOOD property page dialogs for the VVES-related HOOD classes, the addition of customized menus, and the coding of functionality for setting up inter-process communication with the Matlab computation engine. Also a GUI "wizard" was created to guide the user through the process of defining a typical isolation system, which could then be fine-tuned via the normal object property page GUIs.

P517296.PDF [Page: 15 of 50]

2. IMPLEMENTATION

A typical configuration of the engine isolation system considered by the VVES programs [1] consists of the following structural components:

1. an engine block,
2. the first isolator layer,
3. a raft (rigid or beam),
4. a second isolator layer,
5. zero or more additional rafts and isolator layers, and
6. a rigid ground plane.

All of these components were identified to become corresponding classes in the HOOD object oriented design for the engine isolation system. In addition to the structural components, logical components such as attachment points and lumped masses as well as database components such as frequency-dependent material properties were also identified to become HOOD classes. The following subsections discuss in detail each HOOD class used in the VVES model.

2.1 Attachment Points

In the VVES model of the vibration isolation system, the engine, raft and ground are connected by visco-elastic vibration isolator mounts. The connection points are referred as "attachment points" and each of these points is identified by a unique integer number.

Attachment points always exist as child nodes in the scene graph. They belong to an engine, raft, or ground plane. Additional attachment points can be added that are direct children nodes of the scene graph. This type of arrangement is usually used when connecting individual beam elements to isolators. Attachment points are used to connect the objects in the model and thus more than one object will refer to the same attachment point. The attachment point's cross reference list is updated every time an object references the point. If the point is removed, all the affected objects are notified.

P517296.PDF [Page: 16 of 50]

A ground connection point is a special variation of an attachment point. It also keeps track of which degrees of freedom are constrained. This information is used only for the generation of VVES input files and does not have any graphical effect.

2.1.1 Properties

2.1.1.1 Geometric representation

Each attachment point corresponds to a point in three-dimensional space. These points are represented in the software by the AttachmentPoint class. Graphically, the attachment point is seen as a sphere. This appearance is used to ensure appropriate selection of a user desired point for examination or alteration using the mouse.

2.1.1.2 Reference

Attachment points, which are associated with the engine, raft and ground plane, are thought of as "belonging" to another physical object in the model. Thus if an engine is moved in the model, the corresponding attachment points would be expected to move as well since they depend on relative locations on the engine. To achieve this connection, all attachment points have their locations in space defined relative to a reference point. This point is typically the origin of the "parent" object. When the parent object moves, it updates the reference point location for its attachment point children.

2.1.1.3 VVES node number

VVES codes require each node to be identified by a unique integer number. A counter is used to keep track of the number of nodes in a model so that new nodes will be assigned a unique identifier. Another requirement of the VVES codes is that the range of node identifiers must be consecutive. To achieve this, a renumbering of the nodes used in the model is done before the Matlab input files are generated. This guarantees that the node numbers used in the calculations reflect the node numbers shown in the three dimensional model displayed on the screen.

P517296.PDF [Page: 17 of 50]

2.1.1.4 Color and Size

The color and size of all attachment points is shared. If the user changes the color of one point, all the points are changed to the new color. A static Material node (HOOD class used for color) is created once when the first attachment point is created. A reference counter is used to keep track of how many nodes are using this color node. As one attachment point is destroyed, the reference counter is decreased. When the counter reaches zero the color material node can safely be destroyed.

2.1.2 Serialization

The attachment point data stream consists of inherited information from its parent class (SFGNode) followed by the following:

ID:	VVES node identifier
LocalCenter:	coordinates of the point in space relative to the reference point
reference:	global coordinates of the reference point
radius:	radius of sphere for visualization (common to all points)
divisions:	refinement level of sphere. The higher the number, the more polygons are used to represent the sphere (for visualization only)
Center:	global coordinates of the point in space
Color:	RGB color of the attachment point (common to all points)

2.2 Engine

One of the primary objects for any VVES model is the engine. There must be only one engine in a model. In the VVES programs, the engine is modeled as a rigid body, which is characterized by the location of its center of gravity, its mass and mass moments of inertia about three principal directions. However, in the model generator, the dimensions of the engine block are also required for establishment of various geometric relations, automatic calculation of mass moments of inertia and display purposes.

P517296.PDF [Page: 18 of 50]

The origin of the local coordinate system for the engine is defined at the middle of the back bottom edge of the engine block, with the local x'-direction along the "length", local y'-direction along the "width", and local z'-direction pointing upward. With this definition, for an uniform engine block with length "a", width "b" and height "h", the coordinates of the center of gravity are (a/2, 0, h/2). In the present version of the VVES model generator, it is assumed that the local coordinate system, x'y'z', of the engine coincides with the global coordinate system.

2.2.1 Properties

2.2.1.1 Geometric representation

The engine is visualized as a rectangular block. To allow for a more realistic depiction of the engine, an SFGNode (Single Field Graphics Node) member can also be defined. In this case, this member is given control to draw the engine. But this is only used for enhancing the graphical representation of the engine, and has no effect on the engine properties to be used in the VVES analyses.

2.2.1.2 Attachment points

Because the engine is treated as an SFPNode derived class, the functionality inherited from the SFPNode allows the engine class to maintain a list of child nodes. This list is used to store the center of mass (as a node) and its support attachment points. The first child in the list is always the node at the center of mass.

2.2.1.3 Mass and mass moments of inertia

The engine class inherits its inertia properties from the GenericMass class. These properties are its mass and mass moments of inertia about the three local axes: x', y', and z' measured at the center of mass. For a rectangular engine block with uniform mass distribution, the mass moments of inertia at the center of mass about the three axes can be related to the mass and dimensions as:

P517296.PDF [Page: 19 of 50]

$$I_{x'x'} = \frac{\left(width^2 + height^2\right) \cdot mass}{12}$$
$$I_{y'y'} = \frac{\left(length^2 + height^2\right) \cdot mass}{12}$$
$$I_{z'z'} = \frac{\left(width^2 + length^2\right) \cdot mass}{12}$$

where length, width and height indicate dimensions along the local x'-, y'- and z-directions, respectively. Also provided in the model generator is a capability to compute the mass moments of inertia from user supplied mass and radii of gyration as:

$$I_{x'x'} = \left(ROG_{x'}\right)^2 \cdot mass$$
$$I_{y'y'} = \left(ROG_{y'}\right)^2 \cdot mass$$
$$I_{z'z'} = \left(ROG_{z'}\right)^2 \cdot mass$$

where $ROG_{x'}$, $ROG_{y'}$ and $ROG_{z'}$ denote radii of gyration about the three local axes.

2.2.1.4 Constraints

The engine node, which is located at the center of mass of the engine block, has six degrees of freedom, including translations in the global X, Y, and Z directions, and rotations about the global X, Y, and Z axes. Boundary constrains can be applied to any of these degrees of freedom. The DOFCONSTRAINT type and its operators manipulate six bits to keep track of each degree of freedom using only one integer. For example, bit one is used to store the constraint information for translation in the X-direction. If it has a value of one, the constraint has been enabled, otherwise it has not. The second and third bits are for translation in the Y and Z directions, respectively. The next consecutive bits are for rotations about the X, Y, and Z axes.

2.2.1.5 Color

The engine has its own Material node so that the user can customize the color without affecting the color of any other objects in the model.

2.2.2 Serialization

The engine data stream consists of information from its parent class (SFPNode) followed by the following:

Mass:	mass of the engine (kg)
Ixx:	mass moment of inertia about the x'-axis ($kg \cdot m^2$)
Iyy:	mass moment of inertia about the y'-axis ($kg \cdot m^2$)
Izz:	mass moment of inertia about the z'-axis ($kg \cdot m^2$)
Length:	dimension of engine in the x'-direction (graphic representation only)
Width:	dimension of engine in the y'-direction (graphic representation only)
Height:	dimension of engine in the z'-direction (graphic representation only)
Origin:	global coordinates of the mid point of the back bottom edge
Constraints:	bit encoded integer of the constrained degrees of freedom
Color:	RGB color of the engine

2.3 Rigid Raft

A rigid raft is one of the primary objects in a VVES model. A typical VVES model of engine isolation system can contain either one or none rigid raft. However, the present VVES model generator allows for multiple rigid rafts in one model although this is not currently supported by the VVES programs.

The local coordinate system of the rigid raft is defined in the same way as the local system of an engine. The origin of the local system is located at a middle of the back bottom edge. The local x'- and y'-axes are along the length and width directions and the local z'-axis is pointing upward. The directions of the local axes are assumed to coincide with those of the global axes. When the rigid raft is generated through the model wizard, the center of mass of the raft is aligned with the center of mass of the engine. However, this default location of the rigid raft can be changed later if required.

P517296.PDF [Page: 21 of 50]

2.3.1 Properties

2.3.1.1 Geometric representation

A rigid raft is visualized as a rectangular block.

2.3.1.2 Attachment points

A raft is an SFPNode derived class and a rigid raft is derived from the raft class. The functionality inherited from the SFPNode allows the class to maintain a list of child nodes. The raft uses this list to store the center of mass and its upper and lower attachment points. The first child in the list is always the node at the center of mass. Two lists of pointers are used to keep track of the attachment points in the child list belonging to the upper surface and bottom surface of the raft, respectively.

2.3.1.3 Mass and mass moment of inertia

The raft inherits its inertia properties from the GenericMass class. These properties include mass and mass moments of inertia about the three local axes: x', y', and z' measured at the center of mass. In the VVES model generator program, the directions of the local axes are assumed to coincide with those of the global coordinate system. The mass moments of inertia can be input directly, or computed from either user supplied dimensions or radii of gyration, as described earlier in section 2.2.1.3.

2.3.1.4 Constraints

The rigid raft has six degrees of freedom defined at its center of mass. These degrees of freedom includes translations in the global X, Y, and Z directions and rotations about the global X, Y, and Z axes. Boundary constrains can be applied to these degrees of freedom. The DOFCONSTRAINT type and its operators manipulate six bits to keep track of each degree of freedom using only one integer. For example, bit one is used to store the constraint information for translation in the X-direction. If it has a value of one, the constraint has been enabled, otherwise it has not. The

P517296.PDF [Page: 22 of 50]

second and third bits are for translation in the Y and Z directions respectively. The next consecutive bits are for rotations about the X, Y, and Z axes.

2.3.1.5 Color

Each raft has its own Material node so that the user can customize the color without affecting the color of any other object in the model.

2.3.2 Serialization

The Raft data stream consists of information from its parent class (SFPNode) followed by the following:

Mass:	mass of the raft (kg)
Ixx:	mass moment of inertia about the x'-axis ($kg \cdot m^2$)
Iyy:	mass moment of inertia about the y'-axis ($kg \cdot m^2$)
Izz:	mass moment of inertia about the z'-axis ($kg \cdot m^2$)
Length:	dimension of raft in the x'-direction (graphic representation only)
Width:	dimension of raft in the y'-direction (graphic representation only)
Height:	dimension of raft in the z'-direction (graphic representation only)
Origin:	global coordinate of the mid point of the back bottom edge
Color:	RGB color of the raft

The rigid raft data stream consists of inherited information from its parent class (Raft) followed by the following:

Constraints:	bit encoded integer of the constrained degrees of freedom

2.4 Beam Raft

A beam raft is a flexible raft represented by a number of lateral and longitudinal beams. The center lines of all beams are assumed to be in a bounding plane of the beam raft, which is

P517296.PDF [Page: 23 of 50]

rectangular in shape. The origin of the local coordinate system is defined at the mid point of the back edge of the raft bounding plane, similar to local system of the rigid raft. The local x'- and y'-axes are defined along the length and width directions, respectively and the local z'-axis is pointing upward. In the present version of the VVES model generator, the axes of the local system are assumed to coincide with those of the global system, which is also similar to the definition of the local system for the rigid raft. When a beam raft is created using the model wizard, the center of the bounding plane is aligned with the center of mass of the engine. However, this default location of the beam raft can be modified later.

2.4.1 Properties

2.4.1.1 Geometric representation

Although the shape of the bounding plane of a beam raft is rectangular, the actual representation of the raft relies on the actual cross-sectional geometry of the beams that it contains. The beam's cross-section can be any of the following shapes: I-beam, T-beam, Angle beam, Flatbar beam, and rectangular hollow beam. The drawing of the beams is the responsibility of the beam classes that are part of the HOOD core sub-system.

2.4.1.2 Attachment points

A raft is an SFPNode derived class and a beam raft is derived from the raft class. The functionality inherited from the SFPNode allows the class to maintain a list of child nodes. The raft uses this list to store the beam nodes and the attachment points. For consistency with the rigid raft, the first child in the node list is reserved for dummy node at the center of the bounding plane, although this node is not used in the generated VVES input data files. Because, for a beam raft, attachment points are always at the center of beam cross-sections at both ends of beam elements, the upper and lower surfaces of the raft become indistinguishable, and as a result, only the upper list of pointers is used to keep track of the attachment points. Beam nodes are automatically generated at all intersections of lateral and longitudinal beams. In the model wizard, attachment nodes are also automatically generated based on the in-plane coordinates of the attachment nodes on the previous structural component. Attachment points can also be inserted manually.

P517296.PDF [Page: 24 of 50]

2.4.1.3 Beam elements

The center lines of beams in a beam raft are assumed to be along either the lateral (parallel to global Y) or the longitudinal (parallel to global X) direction. The center of the beam cross-section is always assumed to be in the bounding plane of the beam raft and the local y'-axis of the beam cross-section is always coincide with the global Z direction.

Beam elements can also be inserted directly into the model without belonging to a beam raft. In this case, the beam location is defined as by two attachment points. The orientation and cross-section type of the beam element created in this manner can be specified in a similar way as for the isolator. See the discussion of the related isolator properties in sections 2.5.1.2 and 2.5.1.3.

The HOOD system has existing beam classes for I-beam, T-beam, Angle beam, Flatbar beam, and rectangular hollow beam. These classes provide beam cross-section definition and drawing capabilities. The present model generator makes use of these classes.

2.4.1.4 Material

All beam elements in a beam raft are assumed to be made of the same material. However, different material properties can be assigned independently to each beam element which does not belong to the beam raft. The material properties are stored in a separate material database which can be updated by the user.

2.4.1.5 Color

All beam elements share the same Material node for color definition. The user can customize the color for beams without affecting the color of any other objects in the model.

2.4.2 Serialization

The Raft data stream consists of inherited information from its parent class (SFPNode) followed

P517296.PDF [Page: 25 of 50]

by the following:

Mass:	mass of the raft (kg)
Ixx:	mass moment of inertia about the x'-axis ($kg \cdot m^2$)
Iyy:	mass moment of inertia about the y'-axis ($kg \cdot m^2$)
Izz:	mass moment of inertia about the z'-axis ($kg \cdot m^2$)
Length:	dimension of the bounding plane in the x'-direction
Width:	dimension of the bounding plane in the y'-direction
Height:	dimension of the bounding plane in the z'-direction
Origin:	global coordinates of the mid point of the back edge
Color:	RGB color of the raft

Among these data, the mass and mass moment of inertia are not user input and will be set to zero in the VVES input data file, raft.ext.

The Beam Raft data stream consists of the above information inherited from its parent class (Raft) as well as the following unique information:

Material:	name of the material from the material database

The Beam data stream consists of inherited information from its parent class (SFGNode) followed by the following:

Element:	true or false. True if this beam is not part of a beam raft
Origin:	global coordinates of the center of geometry
Length:	length of the beam (m)
DrawingMode:	drawing mode
Orientation:	orientation point (valid only if Element is true)
Attachment1:	node ID
Attachment2:	node ID
Color:	color
BeamMaterial:	name of material property type (valid only if Element is true)

P517296.PDF [Page: 26 of 50]

BeamType: name of beam cross-section type

Lateral: true if this is a lateral beam; false for a longitudinal beam (valid only if Element is false)

2.5 Isolators

Isolators are key elements of VVES models for engine isolation systems. Each isolator contains a flexible visco-elastic component attached to an upper and a lower attachment point. The connections between the upper and lower surfaces of the flexible part and the attachment points are assumed to be rigid.

Two types of isolators can be considered. They are geometrically definable and non-geometrically definable, and are categorized as Type I and Type II isolators, respectively, in the VVES programs. For the first type, geometric shape and dimensions of the isolators are defined in the input data and a linear static finite element analysis is performed internally to determine the condensed stiffness and mass matrices. For the second type, the condensed stiffness and mass matrices are provided by the user, so detailed geometric properties of the isolator are not required in the VVES computations. Presently, these two types of isolators cannot be mixed in one model as this is not yet handled in the VVES codes. Furthermore, the correctness of the non-geometrically defined isolator option in the VVES programs has not been fully verified.

2.5.1 Properties

2.5.1.1 Geometric representation

Three general shapes are provided for geometrically definable isolators, including rectilinear, hollow cone, and solid cone. The required geometric parameters to define these shapes are:

Rectilinear: top length, top width, base length, base width, and height

Hollow cone: top inner diameter, top outer diameter, base inner diameter, base outer diameter, and height

Solid cone: top diameter, base diameter, and height

IR517296.PDF [Page: 27 of 50]

The origin of the local coordinate system of an isolator is located at the lower attachment point. The local z'-axis is along the line connecting the lower and upper attachment points and pointing from the lower point to the upper one. An offset of the bottom surface of the flexible component from the lower attachment point is used to adjust the location of the base of the isolator along the local z'-axis. For rectilinear isolators, the length and width refer to the dimensions in the local x' and y'-directions, respectively. The local y'-z' plane of each isolator is defined by the local z'-axis and an orientation point.

The user may choose either to show or not to show the actual shape of the isolator. In the second case, the isolator is drawn as a simple line connecting the two attachment points. When an isolator does not have both attachment points defined, it is drawn as a sphere.

2.5.1.2 Attachment points

The position of an isolator is fully defined by a pair of upper and lower attachment points. When an attachment point is assigned to the isolator, the isolator adds itself to the attachment point's cross-reference list. Then, if the attachment point is deleted or modified, the isolator will be notified.

2.5.1.3 Orientation

The isolator's local y'-z' plane is defined by the plane consisting of the upper and lower attachment points and an orientation point. The local z'-axis is defined using the vector pointing from the lower attachment point to the upper attachment point and the orientation point should not be on the local z'-axis. In the present version of the VVES GUI program, an unique orientation point is generated for each isolator. These orientation points are also treated as attachment points internally in the GUI program.

The orientation point can be defined in a number of ways. The simplest way is to use an existing node in the VVES model. In this case, a copy of the selected node will be created and used as the orientation point. In the VVES input data files, this point will always be seen as a "dummy" node.

P517296.PDF [Page: 28 of 50]

An alternative method for defining the isolator global orientation is to rotate the local isolator orientation plane, y'-z', about the local z' axis of the isolator to the desired position. The rotation angle, β, is referenced to a "base" orientation plane, which is defined by the local z' axis and one of the axes of the global coordinate system. Once the user selects a global axis and enters a rotation angle, the orientation plane will be determined and a point, which is in the orientation plane, but not on the local z'-axis, will be selected as the orientation point. The equations used to calculate the orientation point for any given global axis and rotation angle are presented in Appendix A.

2.5.1.4 Condensed stiffness and mass matrices

The VVES codes have the capability to compute the condensed stiffness and mass matrices based on a linear finite element approximation of the isolator geometry for the Type I isolators. If this computation is desired, meshing parameters must be specified by the user for generating the finite element model. For the rectilinear isolator, the numbers of divisions along the length (local x'), width (local y') and height (local z') need to be defined. For the solid and hollow cone isolators, numbers of divisions need to be provided for the radial, circumferential, and axial (local z') directions, respectively. For isolators with user supplied condensed stiffness and mass matrices, i.e. the Type II isolators, information for the finite element mesh is not required.

2.5.1.5 Pre-stress

In order to take into account the effect of static loading on the dynamic material properties of the visco-elastic material, a pre-stress calculation capability is provided. This capability involves a scaling of the frequency-dependent complex Young's modulus measured at the undeformed reference state by a scaling factor, which is a function of the isolator geometry, material properties and the pre-stress level. The procedure for calculating the modified dynamics material properties is outlined in Appendix B.

P517296.PDF [Page: 29 of 50]

2.5.1.6 Color

The color of all isolators is shared. If the user changes the color of one isolator, all the isolators are changed to the new color. A static Material node is created once when the first isolator is created. A reference counter is used to keep track of how many isolators are using this color node. As an isolator is destroyed, the reference counter is decreased. When the counter reaches zero, the color material node can safely be destroyed.

2.5.2 Serialization

Description:	description
DrawShape:	draw the true shape
ComputeStiffness:	calculate approximate values for condensed stiffness and mass matrices
Beta:	parameter for material characterization for pre-stress calculation
ShapeFactor:	geometric shape factor
Stress:	static stress on isolator
Eratio:	Young's modulus scaling factor
Density:	mass density
Offset:	offset distance from lower attachment point along local z'-axis
Orientation:	orientation point
UpperAttachment:	node ID
LowerAttachment:	node ID
Color:	color
Material:	material name
4poleData:	4-pole matrices
dynamicStiffnessData:	complex stiffness
KVEMatrix:	condensed stiffness matrix
MVEMatrix:	condensed mass matrix

It should be noted that when node IDs for the upper and lower attachment points are retrieved from a file, a pointer to the corresponding attachment point is required. In order to ensure that the attachment point has already been created by the time its corresponding node ID is referenced, al

P517296.PDF [Page: 30 of 50]

isolators must be serialized after the attachment points.

2.6 Isolator Layers

Isolator layers are groups of isolators that have identical physical characteristics. Some of the VVES programs, such as VVESDYNSTIFF and VVES4POLE, require that all isolators in the same layer have the same frequency dependent stiffness properties.

To ensure that isolators within a layer share the same properties, isolators are added as children to the isolator layer parent. When an isolator is modified, a check is made to see if the parent of the isolator is an isolator layer. If the isolator is determined to belong to a layer, the parent is notified of the change and the modified isolator properties are copied automatically to all the other isolators in the same layer (i.e. the other children).

2.7 Frequency-Dependent Visco-Elastic Material Properties

Every isolator must be associated with a type of frequency-dependent visco-elastic material. The material property data is stored in a material database which can be updated by the user. In the present version of the VVESANALYSIS program, three methods are provided for defining frequency-dependent visco-elastic materials. These methods include experimentally measured material properties, in which complex Young's modulus and Poisson's ratio are specified at a series of frequencies, and material properties generated using the Havrilak-Negami and rational polynomial material models. Material property data defined in any of the above methods can be converted between the shear modulus-bulk modulus (G_1,G_2) format and the Young's modulus-Poisson's ratio (E,μ) format.

2.7.1 Experimentally Measured Properties

The experimentally measured material properties are provided in terms of complex values of the Young's modulus and Poisson's ratio at a series of loading frequencies.

P517296.PDF [Page 31 of 50]

2.7.1.1 Serialization

The experimentally measured material data stream consists of Poisson's ratio at zero Hz and the mass density, which are inherited from the FrequencyDependentMaterial class, and the measured frequency-dependent material data. The data is stored in a list and serialized using the CList class bracketed input and output methods. Each element of the list contains: frequency, real part of the Young's modulus, imaginary part of the Young's modulus, real part of Poisson's ratio, imaginary part of Poisson's ratio, delimited by tabs.

2.7.2 Havrilak-Negami Material Model

The Havrilak-Negami material model is defined by a collection of material parameters. The equations relating these parameters to the shear modulus, G_1, and bulk modulus, G_2, are given by:

$$G_1^*(\omega) = G_{1\infty} + (G_{10} - G_{1\infty})[1 + (i\omega\tau_1)^{\alpha_1}]^{-\beta_1}$$
$$G_2^*(\omega) = G_{2\infty} + (G_{20} - G_{2\infty})[1 + (i\omega\tau_2)^{\alpha_2}]^{-\beta_2}$$

where $G_{1\infty}, G_{10}, G_{2\infty}, G_{20}, \tau_1, \tau_2, \alpha_1, \alpha_2, \beta_1, \beta_2$ are material parameters and ω, the loading frequency.

2.7.2.1 Serialization

The FrequencyDependentMaterial data stream consists of information from its parent class (SFNode) followed by the following:

PoissonsRatio:	Poisson's ratio at zero Hz.
MassDensity:	mass density

The Havrilak-Negami data stream consists of information from its parent class (Frequency DependentMaterial) followed by the following material constants: alpha1, beta1, tau1, G10, G1∞, alpha2, beta2, tau2, G20, G2∞.

2.7.3 Rational Polynomial Material Model

P517296.PDF [Page: 32 of 50]

In the rational polynomial model, frequency-dependent complex material properties are also defined by a collection of material parameters. The equations relating these parameters to the shear modulus, G_1, and bulk modulus, G_2, are given by:

$$G_1^*(\omega) = G_1\left(1 - \sum_{n=1}^{2} \frac{a_{1n}}{i\omega + b_n}\right)$$

$$G_2^*(\omega) = G_2\left(1 - \sum_{n=1}^{2} \frac{a_{2n}}{i\omega + b_n}\right)$$

where $G_1, G_2, a_{11}, a_{12}, a_{21}, a_{22}, b_1, b_2$ are material properties and ω the loading frequency.

2.7.3.1 Serialization

The FrequencyDependentMaterial data stream consists of inherited information from its parent class (SFNode) followed by the following:

PoissonsRatio: Poisson's ratio at zero Hz.
MassDensity: mass density

The rational polynomial data stream consists of inherited information from its parent class (FrequencyDependentMaterial) followed by the following material constants: G1, a11, b1, a12, b2, G2, a21, a22.

2.7.4 Direct Complex Dynamic Stiffness

These matrices are used when the Dynamic Stiffness project type has been selected. In this option, diagonal terms of the complex dynamic stiffness matrices for the layer 1 and layer 2 isolators are provided for a series of loading frequencies. These direct stiffness values are then used to generate the DIRECTSTIFF.EXT file for use with the VVESDYNSTIFF code.

In the VVESDYNSTIFF program, it is assumed that there are only 2 layers of isolators and all the

P517296.PDF [Page: 33 of 50]

isolators in each layer are identical. It should be noted that the directly input material data only contain diagonal terms of the complex stiffness matrix. The other terms are computed internally based on the diagonal stiffness terms and the condensed stiffness matrix discussed earlier in section 2.5.1.4.

2.7.5 4-Pole Parameter Matrices

These frequency-dependent, complex material properties are used when the 4-Pole Parameters project type has been selected. These material data include the 4-pole parameter matrices for motions in the global X, Y and Z-directions for the layer 1 and layer 2 isolators, respectively. The values of these parameter matrices are then used to generate the FPX1.EXT, FPY1.EXT, FPZ1.EXT, FPX2.EXT, FPY2.EXT, and FPZ2.EXT files for use with the VVES4POLE code. For a detailed description of the 4-pole model, see Reference [3].

In the VVES4POLE program, it is assumed that there are only 2 layers of isolators and all the isolators in each layer are identical.

2.8 Lumped Mass

A lumped mass is a simple object that can be added to a model. However, it is not a requirement to have any lumped masses in a VVES model.

2.8.1 Properties

2.8.1.1 Geometric representation

A lumped mass may be represented by a block or a sphere depending on whether or not it has non-zero mass moments of inertia. If the lumped mass has non-zero mass moments of inertia, a rectangular block is used to represent the lumped mass. The dimensions of the block can be determined using the given mass and mass moments of inertia as:

$$length = \sqrt{\frac{6\left(I_{ZZ} + I_{YY} - I_{XX}\right)}{mass}}$$

$$width = \sqrt{\frac{6\left(I_{ZZ} - I_{YY} + I_{XX}\right)}{mass}}$$

$$height = \sqrt{\frac{6\left(-I_{ZZ} + I_{YY} + I_{XX}\right)}{mass}}$$

where an uniform mass distribution has been assumed. The mass moments of inertia are given with respect to the global coordinate system and measured at the center of mass. If a lumped mass just has mass defined, a sphere is shown with a radius derived as:

$$radius = \left(\frac{0.75 \cdot mass}{\pi \cdot density}\right)^{\frac{1}{3}}$$

where the density of concrete is used. It should be noted that this radius is only used for display purposes.

2.8.1.2 Attachment points and rigid link

A lumped mass can be connected to an attachment point through either rigid or flexible links. These two types of connections are treated in slightly different ways in the VVES programs. The user is given the option to specify if the attachment is rigid or flexible.

2.8.1.3 Mass moments of inertia

A lumped mass may be assigned only mass or both mass and mass moments of inertia. In the later case, the mass moments of inertia are defined at the center of mass about the three global axes: X, Y, and Z.

P517296.PDF [Page: 35 of 50]

2.8.1.4 Color

The color of all lumped masses is shared. If the user changes the color of one lumped mass, all the lumped masses are changed to the new color. A static Material node is created once when the first lumped mass is created. A reference counter is used to keep track of how many lumped masses are using this color node. As each lumped mass is destroyed, the reference counter is decreased. When the counter reaches zero the color material node can safely be destroyed.

2.8.2 Serialization

The lumped mass data stream consists of the following:

Description: description
Origin: global coordinates of center of mass
Length: derived length for non-zero inertia about the global X-axis (used for graphics)
Width: derived width for non-zero inertia about the global Y-axis (used for graphics)
Height: derived height for non-zero inertia about the global Z-axis (used for graphics)
Radius: derived radius for non-zero mass with no inertia (used for graphics)
Density: density to be used for showing relative size of mass (default to density of concrete)
Mass: mass (kg)
Ixx: mass moment of inertia about the global X-axis ($kg \cdot m^2$)
Iyy: mass moment of inertia about the global Y-axis ($kg \cdot m^2$)
Izz: mass moment of inertia about the global Z-axis ($kg \cdot m^2$)
RigidlyLinked: true or false depending on the type of connection. True for rigid link and False for flexible link
Attachment: node ID
Color: RGB color of the lumped mass

It should be noted that when node ID of the attachment point is retrieved from a file, a pointer to the corresponding attachment point is required. To ensure that the attachment point has already been created by the time its corresponding node ID is referenced, all isolators must be serialized after the attachment points.

P517296.PDF [Page: 36 of 50]

2.9 Configuration Information (VVESInfo Node)

General project information and configuration parameters used by the VVES codes are stored in the VVESInfo node. The MODVVESDoc class keeps a reference to the VVESInfo node to access the configuration options during various operations.

2.9.1 Properties

2.9.1.1 Project information

Information specific to the project includes: project name, project folder, description, author name, and analysis type. The analysis type is limited to three choices, including "Analysis", "Dynamic Stiffness", and "4-pole Frequency", which refer to the VVESANALYSIS, VVESDYNSTIFF and VVES4POLE programs, respectively. This list of analysis types can be easily extended to include a fourth type, active control analysis, in the future.

Configuration information for the project includes the Matlab folder path and the path of the corresponding VVES codes.

The remaining information stored in this class used to used strictly for the VVES code calculations.

2.9.2 Serialization

The VVESInfo data stream consists of the following:

Author:	author description
Description:	project description
Project:	project name
Code:	either "Analysis", "Dynamic stiffness" or "4-pole"
RunMode:	value of the 'ncase' parameter in the VVES codes

P517296.PDF [Page: 37 of 50]

4PoleRunMode: value of the 'ncase' parameter for the VVES4POLE.m code

eigenvalues: value of the 'NFREQ' parameter in the VVES codes

scale: value of the 'scale1' parameter in the VVES codes

MatlabDirectory: folder containing matlab executable

InputDirectory: project folder

CodeDirectory: folder containing VVES code files

Parametric: boolean variable used to track parameter file generation (feature disabled)

StaticLoading: boolean variable used to indicate if static pre-stress loading is to be calculated

FreeVibration: boolean variable used to indicate if these results are to be calculated

frequencyResponse: boolean variable used to indicate if these results are to be calculated

forceTransmissibility: boolean variable used to indicate if these results are to be calculated

displacementResponse: boolean variable used to indicate if these results are to be calculated

quasiStatic: boolean variable used to indicate if these results are to be calculated

forceTransmissibilityCalculationNode: node ID

forceTransmissibilityLoadPointNode: node ID

forceAppliedAtCOM: boolean variable used to indicate if force applied at engine center of mass

force: applied force

lowFrequency: low frequency of the analysis range

highFrequency: high frequency of the analysis range

frequencyPoints: number of frequency points to calculate at

complexEigenvalues: boolean variable used to indicate if complex eigenvalues (rather than real) are to be calculated

roll: maximum engine roll

pitch: maximum engine pitch

frequencyResponseCalculationNode: node ID

velocityRatio: boolean variable used to indicate if a velocity ratio is to be calculated

velocityNode1: node ID

velocityNode2: node ID

frequencyResponseLoadPointNode: node ID

loadAtNode: boolean variable used to indicate if a load is at a node

loadLocation: location of load (when not at a node)

P317256.PDF [Page: 38 of 50]

loadDOF: degree of freedom of load

supportMotionLocation: location of support motion

displacementResponseCalculationNode: node ID

2.10 VVES Input File Creation

There are a large number of input files that need to be generated before the VVES codes can be executed by Matlab. To simplify the construction of these files, each file has been associated with its own file generation class. These classes are all derived from the 'ofstream' class. For example, the input file ENGINE.EXT is created by the class VVESEngineStream. The class is defined to process an Engine object and produce an output file in the format expected by the VVES codes.

2.11 Model Wizard

To aid in the construction of models, a model wizard can be used. The model wizard guides the user through a sequence of pages collecting information needed to build the model. When the information has been collected, a model is generated. The wizard generates the model from the top down – first the engine is defined, then the isolators beneath the engine, then the raft below the engine until finally the ground plane is defined. The reason for the top down definition rather than a bottom up definition is that information used in the definition of the engine can be used to simplify the definition of the attachment points on the rafts and ground plane beneath the engine.

The model wizard is designed to handle all possible configurations for a model, which may contain either no raft or an arbitrary number of rafts. Property pages are dynamically created and added to the property sheet parent at various stages in the cycle.

2.12 Matlab Server

Matlab provides its own API for inter-application communication. A connection is requested using the 'engOpen' command which starts a new instance of the Matlab server. If a connection is made, the current input directory is added so that Matlab can find the generated script file. A buffer is also supplied to Matlab to keep track of its output messages in case of an error. The

P517296 PDF [Page: 39 of 50]

script file to be evaluated by Matlab includes 'quit' as the final command which will terminate the running instance of Matlab once the calculation have been completed. If the evaluation of the script file is successful, the 'engEvalString' API returns a non-zero result and the user is informed that the processing has been successfully completed. If the result is zero, an error has occurred and the output stored in the buffer is displayed to the user and written to an error log file (err.txt).

P517296.PDF [Page 40 of 50]

3. POST PROCESSING

The results produced from a VVES analysis consists of a collection of text files. The format of these files is detailed in [1].

3.1 Charting

The VVES model generator can chart the data contained in these files using the ChartFX class. This class is a wrapper for the ChartFX ActiveX object and is available by including the ChartFX module in the HOOD application.

Each curve on the chart is referred to as a "series". Output files generated in frequency response, force transmissibility or displacement transmissibility analyses using the VVES code consist of 6 or more series (12 if phase data is written in the file in addition to the 6 degrees of freedom).

A list of charts is made using the ChartFXBrowser object. This object must be part of the current scene graph and must be accessible to the VVESDocument which reads the data files and constructs the charts. A ChartFXBrowser object has been added to the VVESInfo class to meet these requirements.

The data plotted in the charts is available for inspection through the interface. This is a functionality of the ChartFX class.

3.2 Animation

The model can be animated using Matlab. A connection to the Matlab server is established and the 'ShowAnim.m' script is executed. This script creates a new window which shows the animation. The animation runs for 15 seconds and then the window is closed.

P517296.PDF [Page: 41 of 50]

4. TESTING

To test the VVES model generator, all three examples provided in the VVES user's manual [1] were re-created and input files were generated for the VVES codes. The VVES analysis results were then compared with the those provided in the VVES manual. The output files produced using the input files created by the present model generator were identical to the output files produced using the input files given in [1].

The input files created by the VVES model generator did not exactly match the sample input files provided in [1] for the models being tested. These discrepancies were due to the different definition of the global coordinate systems, and the use of additional nodes for describing orientations of the isolators in the present VVES model generator. Both differences are acceptable according to the VVES manual and do not affect the analysis results.

In addition to the three examples mentioned above, various VVES models with beam raft and individual beams have also been generated and tested. For all these models, reasonable results have been obtained from the VVES programs.

The project files for the test examples are included in the VVES model generator installation in the "samples" folder. Files used to specify the frequency dependent isolator properties such as complex stiffness and 4-pole parameters are also included with the corresponding project.

P517296.PDF [Page: 42 of 50]

5. REFERENCES

[1] S.G. HUTTON, "Matlab Computer Programs for Vibration and Control of Vibrations Transmitted from Marine Engines – Users Manual", Vibe-Tek Research Ltd., Vancouver, BC, Canada, March 2000

[2] M. LICHODZIJEWSKI and T. MACADOM, "HOOD User's Manual", Martec Technical Report TR-99-05, February 1999.

[3] S.G. HUTTON, "Optimization of Vibration Mount Properties for Application to Shipboard Diesel Engines", Vibe-Tek Research Ltd., Vancouver, BC, Canada, March 2000

P517296.PDF [Page: 43 of 50]

APPENDIX A: CALCULATION OF ISOLATOR ORIENTATION PLANE FROM USER DEFINED GLOBAL AXIS AND ROTATION ANGLE

Consider an isolator connected between two attachment points with global coordinates as:

$$\bar{X}_1 = \{x_1, y_1, z_1\}^T, \quad \bar{X}_2 = \{x_2, y_2, z_2\}^T. \tag{A.1}$$

The local z-axis can be readily defined by:

$$\bar{z}' = \begin{Bmatrix} (x_2 - x_1)/r \\ (y_2 - y_1)/r \\ (z_2 - z_1)/r \end{Bmatrix} = \begin{Bmatrix} l \\ m \\ n \end{Bmatrix} \tag{A.2}$$

where

$$r = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2 + (z_2 - z_1)^2}$$

In order to obtain the orientation point, a reference point is first defined in the reference plane, which is the plane formed by the local z' axis and the user selected global axis, and but not on the local z' axis. If the global x axis is selected and the local z' is not perpendicular to the x axis, the reference point can be readily computed by:

$$\{x_R, y_R, z_R\} = \left\{ \frac{l \cdot (x_2 - x_1) + m \cdot (y_2 - y_1) + n \cdot (z_2 - z_1)}{l}, 0, 0 \right\} \quad \text{for } l \neq 0 \tag{A.3a}$$

If the local z' direction is perpendicular to the global x, we use:

$$\{x_R, y_R, z_R\} = \{r,\ y_2 - y_1,\ z_2 - z_1\} \text{ for } l = 0 \tag{A.3b}$$

Similarly, if global y is selected, we have:

$$\{x_R, y_R, z_R\} = \left\{ 0, \frac{l \cdot (x_2 - x_1) + m \cdot (y_2 - y_1) + n \cdot (z_2 - z_1)}{m}, 0 \right\} \quad \text{for } m \neq 0 \tag{A.4a}$$

P517296.PDF [Page: 44 of 50]

and

$$\{x_R, y_R, z_R\} = \{x_2 - x_1,\ r,\ z_2 - z_1\} \text{ for } m = 0 \tag{A.4b}$$

depending on whether the local z' axis is perpendicular to the global y axis. If global z is selected by the user, we then have:

$$\{x_R, y_R, z_R\} = \left\{0, 0, \frac{l \cdot (x_2 - x_1) + m \cdot (y_2 - y_1) + n \cdot (z_2 - z_1)}{n}\right\} \text{ for } n \neq 0 \tag{A.5a}$$

and

$$\{x_R, y_R, z_R\} = \{x_2 - x_1,\ y_2 - y_1,\ r\} \text{ for } n = 0 \tag{A.5b}$$

With the reference point established, the reference vector can be constructed, which is pointing from the upper attachment point to the reference point defined above as:

$$\Delta\bar{Q} = f \cdot \begin{Bmatrix} x_R - (x_2 - x_1) \\ y_R - (y_2 - y_1) \\ z_R - (z_2 - z_1) \end{Bmatrix} \tag{A.6}$$

where f is a scaling factor

$$f = \frac{r}{\sqrt{(x_R - (x_2 - x_1))^2 + (y_R - (y_2 - y_1))^2 + (z_R - (z_2 - z_1))^2}} \tag{A.7}$$

which is used to ensure that the length of the reference vector is identical to the distance between the attachment points.

For a rotation of angle β about the local z' axis, the rotation matrix can be formulated as:

$$\boldsymbol{R} = \boldsymbol{I} + \frac{\sin\beta}{\beta} \cdot \boldsymbol{S}(\vec{\beta}) + \frac{1 - \cos\beta}{\beta^2} \cdot \boldsymbol{S}(\vec{\beta}) \cdot \boldsymbol{S}(\vec{\beta}) \tag{A.8}$$

P517296.PDF [Page: 45 of 50]

where **S** is a skew-symmetric matrix defined by:

$$\boldsymbol{S}(\vec{\beta}) = \begin{bmatrix} 0 & -\beta_3 & \beta_2 \\ \beta_3 & 0 & -\beta_1 \\ -\beta_2 & \beta_2 & 0 \end{bmatrix} \tag{A.9}$$

and β_1, β_2 and β_3 form a pseudo-vector as:

$$\vec{\beta} = \{\beta_1, \beta_2, \beta_3\} = \{\beta \cdot l, \beta \cdot m, \beta \cdot n\}. \tag{A.10}$$

Using the reference vector and the rotation matrix, the global coordinates of the orientation point can be computed as:

$$\vec{x}_3 = \{x_3, y_3, z_3\} = \vec{x}_2 + \boldsymbol{R}\,\Delta\vec{Q}\,. \tag{A.11}$$

P517296.PDF [Page: 46 of 50]

APPENDIX B: CALCULATION OF SCALING FACTOR OF YOUNG'S MODULUS FOR TAKING INTO ACCOUNT THE PRE-STRESS EFFECT

Step 1: Calculate the shape factor of the isolator S, which is the ratio of the areas of the loaded and unloaded surfaces. For isolators of cylindrical shape, the shape factor is defined as:

$$S = \frac{D}{4H} \tag{B.1}$$

where D and H denote the diameter and height of the cylinder, respectively. For rectilinear isolators,

$$S = \frac{LW}{2H(L+W)} \tag{B.2}$$

where L, W and H indicate length, width and height of the isolator. To obtain the above formulas for S, it has been assumed that load is applied in the vertical (height) direction.

Step 2: Calculate the apparent modulus E_a.

$$E_a = E \cdot (1 + \beta \cdot S^2) \tag{B.3}$$

where E is Young's modulus at low strain and at a frequency of zero Hz, and β is a parameter corresponding to the material characterization of the isolator ($\beta = 2$ for unfilled polymers, $1 < \beta < 2$ for carbon black filled polymers).

Step 3: Calculate the stretch. Assume σ is the averaged static compressive stress in the isolator. Using the formula of Gent and Lindley for constrained compression of rubber block, given by:

$$\sigma = \frac{E_a}{3}\left(\frac{1}{\lambda^3} - \lambda\right), \tag{B.4}$$

P517296.PDF [Page: 47 of 50]

the stretch can be readily solved as:

$$\lambda = \frac{A}{2 \cdot E_a} + \frac{2\sigma^2}{E_a A} - \frac{\sigma}{E_a} \tag{B.5}$$

where

$$A = \left(4E_a^{\,3} - 8\sigma^3 + 4 \cdot E_a^{\,2} \sqrt{E_a^{\,2} - \frac{4 \cdot \sigma^3}{E_a}} \right)^{\frac{1}{3}}. \tag{B.6}$$

Step 4: Calculate the scaling factor. From the Gent and Lindley's formula, the tangent modulus corresponding to a given stretch value can be obtained by differentiation with respect to λ as:

$$E_T = -\frac{\partial \sigma}{\partial \varepsilon} = -\frac{\partial \sigma}{\partial \lambda}. \tag{B.7}$$

As a result, the scaling factor can be obtained as:

$$f = \frac{E_T}{E_a} = \frac{1}{3} \cdot \left(\frac{2}{\lambda^3} + 1 \right) \tag{B.8}$$

Step 5: Calculate the modified frequency-dependent complex Young's modulus.

$$\tilde{E}^*(\omega) = f\, E^*(\omega) \tag{B.9}$$

where E and $\tilde{E}$ indicate original and modified Young's modulus, respectively and * denotes complex variables.

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

MARTEC Limited
1888 Brunswick Street, Suite 400,
Halifax, Nova Scotia, Canada, B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable)

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

Development of a Model Generator and Graphical User Interface for the VVES Programs

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

Polak, M.F., Jiang, L., MacAdam, T.A., Chernuka, M.W.

5. DATE OF PUBLICATION (month and year of publication of document)	6a. NO. OF PAGES (total containing information. Include Annexes, Appendices, etc)	6b. NO. OF REFS (total cited in document)
January 2002	39	2

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).

CONTRACTOR REPORT

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).	9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written)
Project 1gc12	W7707- 008286/001/HAL

10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)	10b. OTHER DOCUMENT NOs (Any other numbers which may be assigned this document either by the originator or by the sponsor.)
Martec Report No. TR-01-20	DREA CR 2001-143

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected)

Unlimited

UNCLASSIFIED
SECURITY CLASSIFICATION OF FOR

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

The VVES (Vibration of Visco-elastic and Elastic Structures) suite of programs are a group of Matlab codes for analysis and design of engine isolation systems composed of engine, raft and ground which are connected by visco-elastic isolators. Among these programs, VVESANALYSIS allows the dynamic characteristics of isolators to be described by frequency-dependent complex material properties. The VVESDYNSTIFF and VVES4POLE require input of the directly measured dynamic stiffnesses and the 4 pole parameter matrices, respectively.

In order to simplify the construction of input files and provide an alternate method to view the VVES analysis results, a model generator and graphical user interface for the VVES programs has been recently developed. This model generator and GUI is developed for the Windows environment using the HOOD toolkit. Details of this development are documented in this report.The reader of this report is assumed to have a reasonable familiarity with the VVES programs and the HOOD's core components.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

diesel engine vibration
vibration isolation
resilient suspension
visco elasticity

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM